Current Trends in Technology and Sciences
ISSN: 2279-0535. Volume: 1, Issue: 3, (Nov. 2012)

CFD Analysis of Cavity Based Combustion of Hydrogen at Mach Number 1.4

Oveepsa Chakraborty
PG Student, Dept of Mech. Engg. NIT, Silchar, Assam, India, oveepsac@gmail.com

Deepak Sharma
PG Student, Dept. of Mech. Engg. NIT, Silchar, Assam, India, deep.cool.u@gmail.com

K. Obula Reddy
PG Student, Dept. of Mech. Engg. NIT, Silchar, Assam, India, obulareddy.bec10@gmail.com

K. M Pandey
Professor, N.I.T Silchar, Assam, India, kmpandey2001@yahoo.com

Abstract - This work involves an application of computational fluid dynamics to a problem associated with the flow in the combustor region of a supersonic combustion ramjet engine (Scramjet). The CFD analysis of the combustion process of a scramjet engine having wall injector at different position on the wall of the combustion nozzle with single cavity and double cavity for L/D ratio 10. The main objective of this work is to design the combustion chamber model by using GAMBIT software, study the combustion processes of Air-Fuel (H2) mixture for the wall injector models with inlet air at Mach number 1.4 and inlet fuel at Mach number 1.4. There are several key issues that must be considered in the design of an efficient combustion chamber of a rocket engine. The main objective of this analysis is to compare the various two-dimensional cavity based models. Numerical results are obtained with the FLUENT software shows that duel cavity based combustor model is found to have good overall agreement with results obtained from literature reviews. To delineate the purely fluid dynamic effects, the flow is treated as non-reacting. The grid independent test was also carried out for better accurate results. The various profiles of static pressure, static temperature at various locations of the flow field are presented. Some discrepancies were observed for static pressure and static temperature in the vicinity of the jets due to unsteadiness in the shock system. The both, air intake and the Hydrogen injection are at same Mach speed. It is observed that a maximum temperature of 1340K can be achieved with injection of Hydrogen at mach 1.4 speed and two cavity nozzle of combustion.

Keywords - wall injector, Mach number, scramjet, static temperature, static pressure, kinetic energy.

Nomenclature:

k = turbulence kinetic energy

 = dissipation rate

ω = specific dissipation rate

G_k = generation of turbulence kinetic energy due to mean velocity gradients

u_j = j_{th} Cartesian component of the instantaneous velocity

ρ = density of fluid

p = static pressure,

τ_{ij} = viscous stress tensor

f_i = the body force

h = static enthalpy

G_b = generation of turbulent kinetic energy that arises due to buoyancy

Y_M = represents the fluctuating dilation

S_ε, S_k = source terms defined by the user.

$C_{1\varepsilon}$, $C_{2}\varepsilon$, $C\mu$ = constants that have been determined experimentally

σ_k, σ_ε = turbulent Prandtl numbers for the turbulent kinetic energy, its dissipation rate.

S = mean rate-of-strain tensor.

1. Introduction

The scramjet engine is one of the most promising propulsive systems for future hypersonic vehicles. Over the last fifty years the scramjet engine technology has been intensively investigated and several such engines have been flight-tested in recent years (Neal, Michael, & Allan, 2005; Paul, Vincent, Luat, & Jeryl, 2004). Research on supersonic combustion technologies is of great significance for the design of the engine and many researchers pay significant attention to the hypersonic air breathing propulsion. The mixing and diffusive combustion of fuel and air in conventional scramjet engines take place simultaneously in the combustor (Huang, Qin, Luo, & Wang, 2010). Since the incoming supersonic flow can stay in the combustor only for a very short period of time, i.e. of the order of milliseconds (Aso, Inoue, Yamaguchi, & Tani, 2009; Huang *et al.*, 2010; Hyungseok, Hui, Jaewoo, & Yunghwan, 2009), and the whole process of combustion has to be completed within this short duration, this is a significant restriction to the design of the scramjet engine. In order to solve this

problem, hydrogen, one of the most promising fuels for the air breathing engine with ~10 times faster reaction than hydrocarbons, is widely used in the scramjet combustor. In recent years, a cavity flame holder, which is an integrated fuel injection/flame-holding approach, has been proposed as a new concept for flame holding and stabilization in supersonic combustors (Alejandro, Joseph, & Viswanath, 2010; Chadwick *et al.*, 2005; Chadwick, Sulabh, & James, 2007; Daniel & James, 2009; Gu, Chen, & Chang, 2009; Jeong, O'Byrne, Jeung, & Houwong, 2008; Kyung, Seung, & Cho, 2004; Sun, Geng, Liang, & Wang, 2009; Vikramaditya & Kurian, 2009). The presence of a cavity on an aerodynamic surface could have a significant impact on the flow surrounding it. The flow field inside a cavity flame holder is characterized by the recirculation flow that increases the residence time of the fluid entering the cavity, and the cavity flame provides a source of heat and radicals to ignite and stabilize the combustion in the core flow.

However, so far, the flow field in the scramjet combustor with multiple cavity flame holders has been rarely discussed, and this is an important issue as it can provide some useful guidance for the further design of the scramjet combustor. Multi-cavity flame holder can produce larger drag forces on the scramjet combustor, as well as improve the combustion efficiency of the combustor. A balance between these two aspects will be very important in the future design of the propulsion system in hypersonic vehicles. At the same time, the combustor configuration, i.e. the divergence angle of each stage, makes a large difference to the performance of the combustor. Researchers have shown that (Huang, Li, Wu, & Wang, 2009) the effect of the divergence angles of the posterior stages on the performance of the scramjet combustor is the most important, and the effect of the divergence angle on the first stage is the least important. When the location of the fuel injection moves forward, the effect of the divergence angle of the former stages becomes more important. In this chapter, the two-dimensional coupled implicit Reynolds Averaged Navier-Stokes (RANS) equations, the standard k-ε turbulence model (Huang & Wang, 2009; Launder & Spalding, 1974) and the finite-rate/eddy-dissipation reaction model (Nardo, Calchetti, Mongiello, Giammartini, & Rufoloni, 2009) have been employed to investigate the effect of the location of the fuel injection on the combustion flow field of a typical hydrogen-fueled scramjet combustor with multi-cavities.

1.1 Cavity Flame Holders

Cavity flame holders create a subsonic region for a recirculation trapped vortex to exist. Fuel and air can be entrained from the free stream flow into the cavity through the shear layer or directly injected into the cavity itself. The trapped vortex concept has been shown by Ben-Yakar and Hanson (1998) to be an excellent method of providing a stable flame holding device. However, it is this same stability which limits the amount of turbulent mass entrainment with the free stream flow.

Cavities are normally classified as open or closed. An open cavity is one in which the shear layer separates at the cavity leading edge and reattaches on the aft edge. A closed cavity is one in which the shear layer is unable to reattach to the aft end of the cavity, and thus attaches to the cavity floor instead. Figure 4 illustrates the geometric classification of cavities. Nestler et al. (1968) determined that the length to depth ratio separating open and closed cavities was approximately 10.

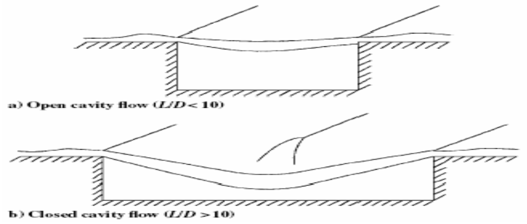

Fig.4. Two general classifications of cavities a) open cavity and b) closed cavity (Gruber et al., 2001; used without permission).

2. LITERATURE SURVEY

Team, Head, Hypersonic Propulsion Division and Dy.Project Director, HSTDV, DRDL, Hyderabad [1] work on Scramjet combustor development and their findings are as following. Among the three critical components of the scramjet engine, the combustor presents the most formidable problems. The complex phenomenon of supersonic combustion involves turbulent mixing, shock interaction and heat release in supersonic flow. The flow field within the combustor of scramjet engine is very complex and poses a considerable challenge in design and development of a supersonic combustor with an optimized geometry. The designer should keep in mind the following goals namely, Good and rapid fuel air mixing, Minimization of total pressure loss, High combustion efficiency.

Ashim Dutta, Zhiyao Yin, Igor V. Adamovich[2] work on “Cavity ignition and flame holding of ethylene–air and hydrogen–air flows by a repetitively pulsed nanosecond discharge” their finding are as following Repetitive nanosecond pulse plasma assisted ignition and flame holding of premixed and non-premixed ethylene–air and hydrogen–air flows are studied in a cavity flows at a pressure of 0.2 atmosphere and the flow velocities of up to 100 m/s. The ignition between fuel and air occurs via formation of multiple filaments in the fuel–air plasma, but although air plasma remains diffuse until the fuel is added. After the ignition occurs in the cavity, during with ignition delay time of a few milliseconds, during this time period the plasma becomes diffuse and the flame couples out to the main flow. The use of a short cavity (length-to-depth ratio L/D = 1) results in repetitive ignition and flame blow-off, caused by slow mixing between the main flows and the cavity. Increasing the length-to-depth ratio to L/D = 3, as well as choking inlet air and fuel flows resulted in stable flame holding and nearly complete combustion in both premixed and non-premixed

ethylene–air and hydrogen–air flows at u = 35–100 m/s. Air plasma temperature before fuel is added ranges from 70 C to 200 C.

Chadwick C. Rasmussena, James F. Driscoll, Kuang-Yu Hsub, Jeffrey M. Donbar, Mark R. Gruber, Campbell D. Carter [3] work on “Stability limits of cavity-stabilized flame in supersonic flows” their findings are as following. Experiments were performed to examine the stability of hydrocarbon-fueled flame in cavity flame holders in supersonic air flow. Methane and ethylene were burned in two different cavity configurations having aft walls ramped at 22.5 and 90. Air stagnation temperatures were 590 K at Mach 2 and 640 K at Mach3. Lean blow out limits showed dependence on the air mass flow rates, cavity geometry, fuel injection scheme, Mach number, and fuel type. But here large differences were noted between cavity floor and cavity ramp injection schemes. Visual observations, planar laser-induced fluorescence of nitric oxide, and shadowgraph imaging were used to investigate these phenomena and the Cavity ramp injection provided better performance.

Daniel J. Micka , James F. Driscoll [4] work on “Combustion characteristics of a dual-mode scramjet combustor with cavity flame holder” their findings are as following. Combustion characteristics of a laboratory dual-mode ramjet/scramjet combustor were studied experimentally. The combustor consists of a sonic fuel jet injected into the supersonic cross flows upstream of a wall cavity pilot flame. These fundamental components are contained in many dual-mode combustor designs. Experiments were performed with an isolator entrance of Mach number 2.2. The stagnation temperature of air has varied from 1040 K to 1490 K, and which correspond to the flight Mach numbers of 4.3–5.4. Both pure hydro- gen and a mixture of hydrogen and ethylene fuels were used. The high speed imaging of the flame luminosity was performed along with measurements of the isolator and combustor wall pressures and also analyzes the various measurements. For the mode of ramjet operation, there are two distinct combustion stabilization locations were found for fuel injection with a sufficient distance upstream of the cavity. At low stagnation temperatures, the combustion was anchored at the leading edge of the cavity by heat release in the cavity shear layer, but at high stagnation temperature, the combustion was stabilized a short distance downstream of the fuel injection jet in the jet-wake. For an intermediate range of temperature, the reaction zone oscillated between the jet-wake and cavity stabilization locations, and the Wall pressure measurements showed that cavity stabilized combustion was the steadiest.

R. W. Pitz , M. D. Lahr, Z. W. Douglas, J. A. Wehrmeyer and S. Hu [5] work on “Hydroxyl Tagging Velocimetry in a Mach 2 Flow with a Wall Cavity” their finding are as following. Hydroxyl tagging velocimetry (HTV) measurements of velocity were made in a Mach 2 flow with a wall cavity. In the HTV method, ArF excimer laser (193 nm) beams pass through a humid gas and dissociate H2O into H + OH to form a tagging grid of OH molecules. In this study, a 7x7 grid of hydroxyl (OH) molecules is tracked by planar laser-induced fluorescence. The grid motion over a fixed time delay yields about 50 velocity vectors of the two-dimensional flow. Instantaneous, single-shot measurements of two-dimensional flow patterns were made in the non-reacting Mach 2 flow with a wall cavity under low and high pressure conditions. The single-shot profiles were analyzed to yield mean and RMS velocity profiles in the Mach 2 non-reacting flow. From this work, the HTV method is applied to a Mach 2 flow with a wall cavity to obtain instantaneous two dimensional velocity images, mean velocity profiles and rms velocity profiles. Velocity measurements are made using HTV in the free-stream and the cavity of the Mach 2 cavity-piloted combustor.

3. CFD MODELING

Computational Fluid Dynamics (CFD) packages are very powerful tools for analyzing any type of fluid flow. They are capable of calculating a large number of flow parameters that are often difficult or impossible to determine experimentally. For optimization purposes, they allow easy manipulation of geometry and flow conditions.

3.1. Solution Methodology and Governing Equations

CFD-ACE uses a control volume approach in calculating flow parameters. The region of interest in the flow simulation of any computational domain is divided that domain into a grid. But in the numerical simulation, each grid element is considered as a control volume that means the properties are constant over its volume. For each of the control volume, fluid flow is simulated by numerically solving partial differential equations that govern the transport of flow quantities, also known as flow variables this will be go through the discretization of partial differential equation. The variables include mass, momentum, energy, turbulence quantities, and mixture fractions and species concentrations. The variables for which transport equations have to be solved will depend on the nature of the flow problem.

The three equations common to all fluid dynamics problems are the conservation of mass, momentum and the energy equations. In differential form these are:

Conservation of mass:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j) = 0$$

Where u_j is the j_{th} Cartesian component of the instantaneous velocity and ρ is the fluid density.

Conservation of momentum:

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_k} + \frac{\partial \tau_{ij}}{\partial x_i} + \rho f_i$$

Here p is the static pressure, τ_{ij} is the viscous stress tensor and f_i is the body force.

Current Trends in Technology and Sciences
ISSN: 2279-0535. Volume: 1, Issue: 3, (Nov. 2012)

Conservation of energy:

$$\frac{\partial}{\partial t}(\rho h)+\frac{\partial}{\partial x_j}(\rho u_j h) = -\frac{\partial q_j}{\partial x_j}+\frac{\partial \rho}{\partial t}+u_j\frac{\partial p}{\partial x_j}+\tau_{ij}\frac{\partial u_i}{\partial x_{ji}}$$

Where q_j is the j-component of the heat flux and h is the static enthalpy.

The three Partial Differential Equations (PDE's), and with any other equations as per dependant on the specific flow problem, now all the governing equations of the flow are discredited on the computational grid, then set of algebraic equations are formed in terms of flow quintiles also known as flow variables that we have to use for specifying that particular flow, and finally the solution of the algebraic equations is determined with the CFD program called as FLUENT. And this method generates the flow variables at each grid point for every simulation and yields the accurate results for that particular flow.

An iterative solution scheme is used by CFD software, FLUENT to solve the algebraic equations. The all discredited algebraic equations are solved sequentially and repeatedly with the goal of improving the solution at each iteration as per the development of processing for the solver. The solution is monitored by viewing global residuals (the difference between the current and previous solution average over the entire domain). A solution is generally considered "converged" when the residuals have decrease by 4-5 orders of magnitude. The most important point to consider when using CFD (or any CFD program) is that the quality of its output is only as good as the quality of its input so care has to be taken to make sure that inputs, such as boundary conditions, fluid properties and fluid models are as accurate for (or applicable to) the specific problem, as far as possible.

3.2. Turbulence Models

The turbulence model used for the CFD models analyzed in this project was the standard k-ε model. It was used because it is well known and applicable to high Reynolds number flows. This original model was initially proposed by Launder and Spalding (1972). For this model the transport equation for turbulent kinetic energy, *k* is derived from the exact equation, but the transport for the dissipation rate, ε was obtained using physical reasoning and is therefore similar to the mathematically derived transport equation of *k*, but is not exact. The turbulent kinetic energy *k*, and its rate of dissipation ε, for this model are obtained by the following equations.

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left\{\left[\mu+\frac{\mu_t}{\sigma_k}\right]\frac{\partial k}{\partial x_j}\right\}+G_k+G_b-\rho\varepsilon - Y_M + S_k$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left\{\left[\mu+\frac{\mu_t}{\sigma_\varepsilon}\right]\frac{\partial \varepsilon}{\partial x_j}\right\} + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\epsilon$$

Where G_k represents the generation of turbulent kinetic energy that arises due to the development of mean Velocity gradients in the flow dynamics, G_b is the generation of turbulent kinetic energy that arises due to the dynamics of buoyancy forces, where as Y_M represents the fluctuating dilation in compressible turbulence that contributes to the overall dissipation rate. S_ε and S_k are source terms defined by the user.

$C_{1\varepsilon}$, $C2\varepsilon$ and $C\mu$ are constants that have been determined experimentally and are taken to have the following values;

$C_{1\varepsilon}$ =1.44, C2ε =1.92, C_μ =0.09

σ_k and σ_ε are turbulent Prandtl numbers for the turbulent kinetic energy and its dissipation rate. These have also been derived experimentally and are defined as follows.

σ_k =1.0, σ_ε =1.3

The turbulent (or eddy) viscosity in the flow field at each point is defined in terms of the local values of turbulent kinetic energy and the dissipation rate is expressed by;

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

Where C_μ is constant and defined above.

The term for the production of turbulent kinetic energy G_k is common in many of the turbulence models studied and is defined as in terms of the some parameters as shown below.

$$G_k = -\rho u_i' u_j' \frac{\partial u_j}{\partial x_i}$$

The modulus of mean rate-of-strain tensor, *S* is defined as

$$S = \sqrt{2S_{ij}S_{ij}}$$

The generation of turbulent kinetic energy that arises due to buoyancy, G_b is defined as follows, and is expressed in various parameters, that govern or relates to the turbulent kinetic energy with respect to the buoyancy.

$$G_b = \beta g_i \frac{\mu_t}{Pr_t}\frac{\partial T}{\partial x_i}$$

As in our present study uses relatively low velocities, the dilation dissipation term, Y_M which accounts for turbulence from compressibility effects is defined as

$$Y_M = 2\rho\varepsilon M_t^2$$

3.3. Combustion Modeling

In a multi-reaction environment, the challenge is often to define the minimum number of reactions necessary to represent the important characteristics of the flame. Here on the other hand an extensive set of reactions is used in order to resolve all the important intermediate species and free radicals as they play an important role in the ignition/extinction mechanism. Combustion mechanism

Current Trends in Technology and Sciences
ISSN: 2279-0535. Volume: 1, Issue: 3, (Nov. 2012)

used to model hydrogen combustion. Nitrogen gas (N2) is inert and therefore does not participate in any reaction, although it affects the rates of some reactions acting as a third body. Kinetic rate of change of a species is described by an Arrhenius rate expression. The source term for each species is ultimately determined from the summation of the change in that particular species from all contributing reactions. Other forms may be more appropriate depending upon the reaction. It is quite common to have concentration dependencies for gas species other than those involved in the reaction (Javed and Chakraborty, 2006).

4. CFD MODEL ANALYSIS

Different profiles are made in GAMBIT and inserted suitable boundaries. Two dimensional meshing are also done in GAMBIT with suitable spacing based on Grid independent test, and the mesh examination has done as per the three main quality tests namely Element aspect ratio, equiangle skew ratio and stretch. The simulation is performed with various possible with options for interactive or batch processing and distributed processing. The various contours are presented along the combustor length. Post-Process the Simulation to get the Results, post process involves the Contours of static pressure and total temperature are seen for the wall injector with the length along the direction of flow. Plots are being drawn between pressure variation and length of wall injector as well as between temperature variation and length of wall injector.

5. RESULT

The pressure and temperature analysis of the nozzle for cavity the changes are seen near the fuel inlet. Various CFD contours of static temperature, velocity contours and static pressure contours for different models are as shown below at Mach number 1.4. In first case we evolutes the various contours with wall injector cavity, with fuel inlet at the bottom of the wall, and the next contours are for the second case, with two cavities at the bottom of the wall.

5.1 Wall injector with cavity (fuel inlet at the bottom wall)

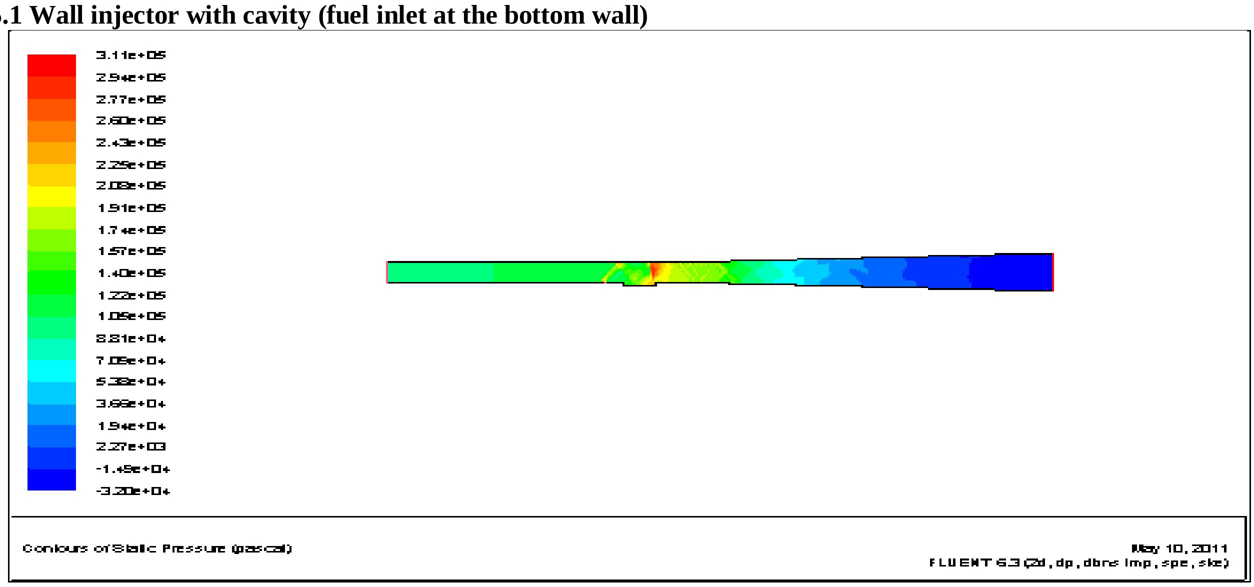

Fig.1. Contours of static pressure (Pascal)

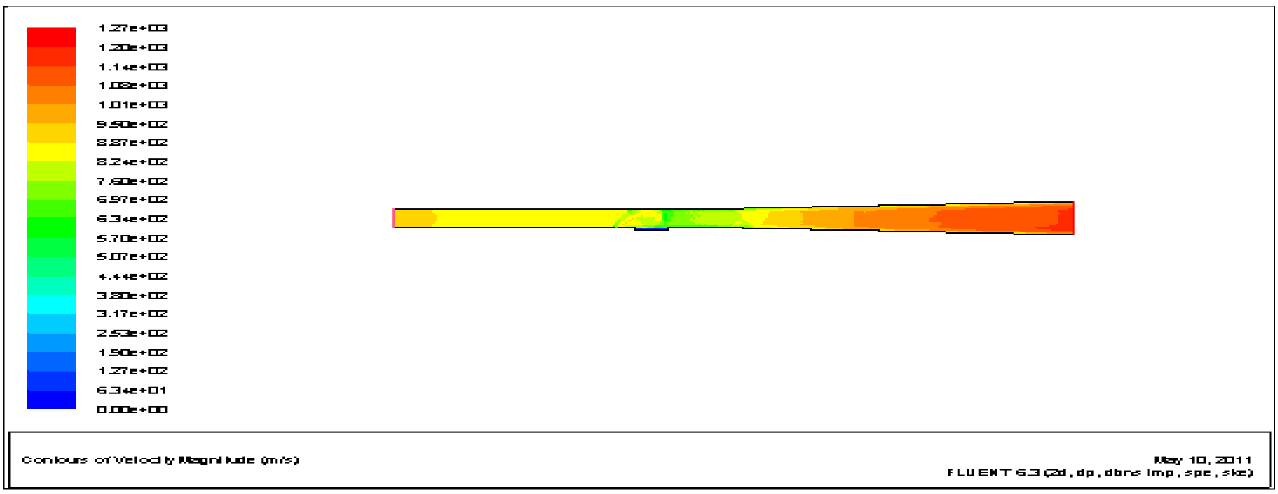

Fig.2. Contours of velocity (m/s)

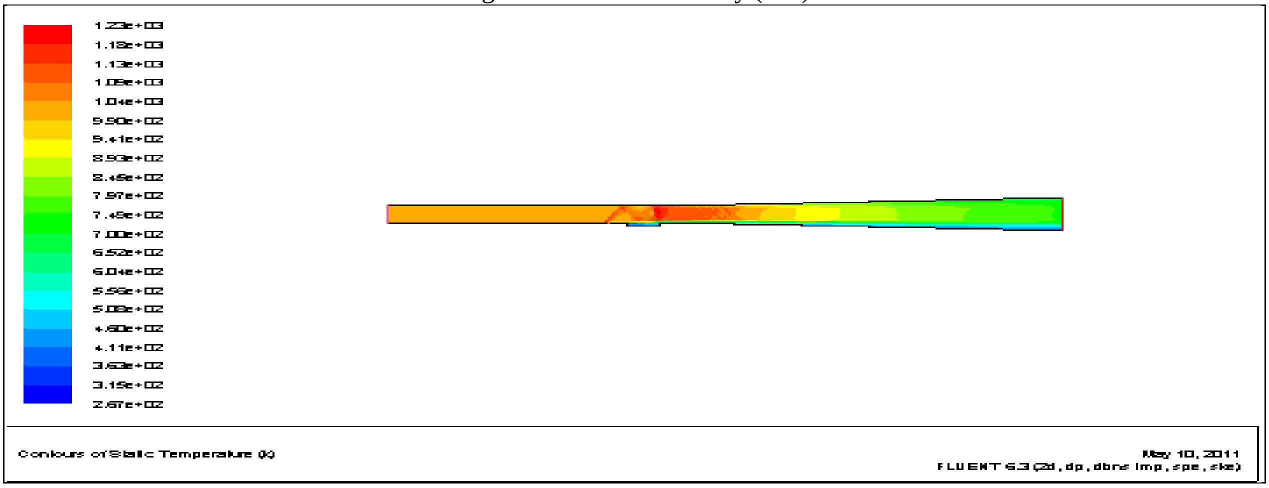

Fig.3. Contours of static temperature (K)

5.2 Wall injector with two cavities (fuel inlet at the bottom wall)

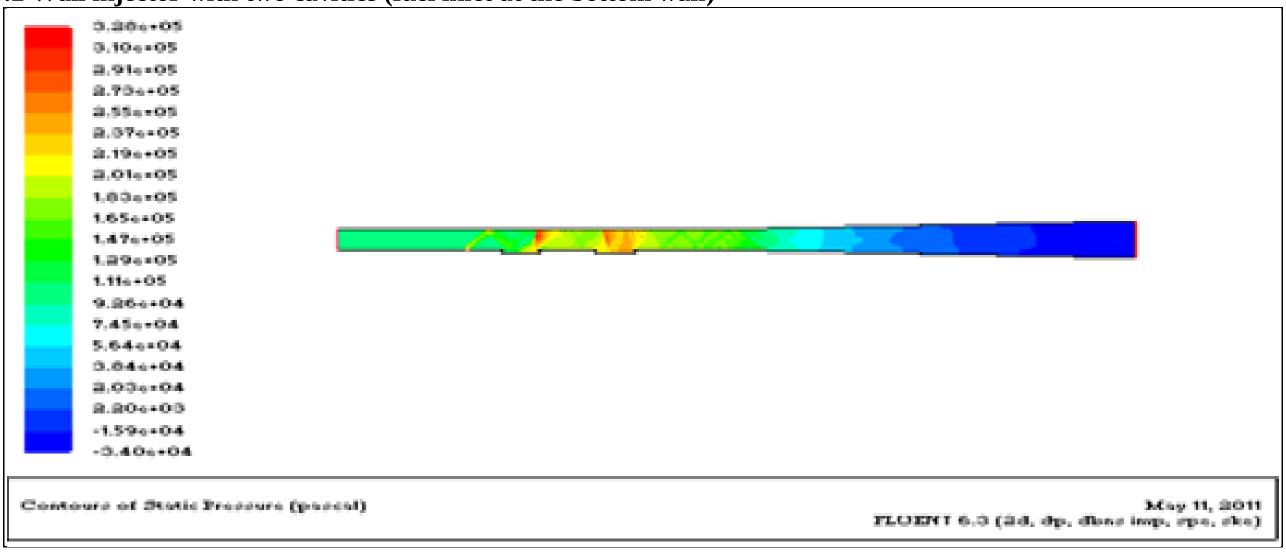

Fig.4. Contours of static pressure (Pascal)

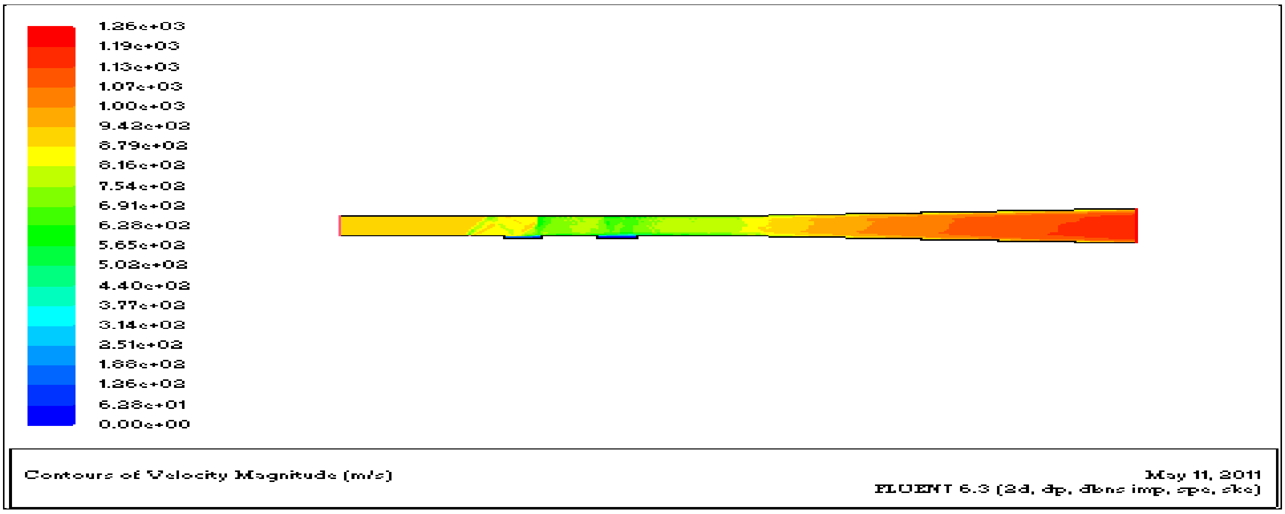

Fig.5. Contours of velocity (m/s)

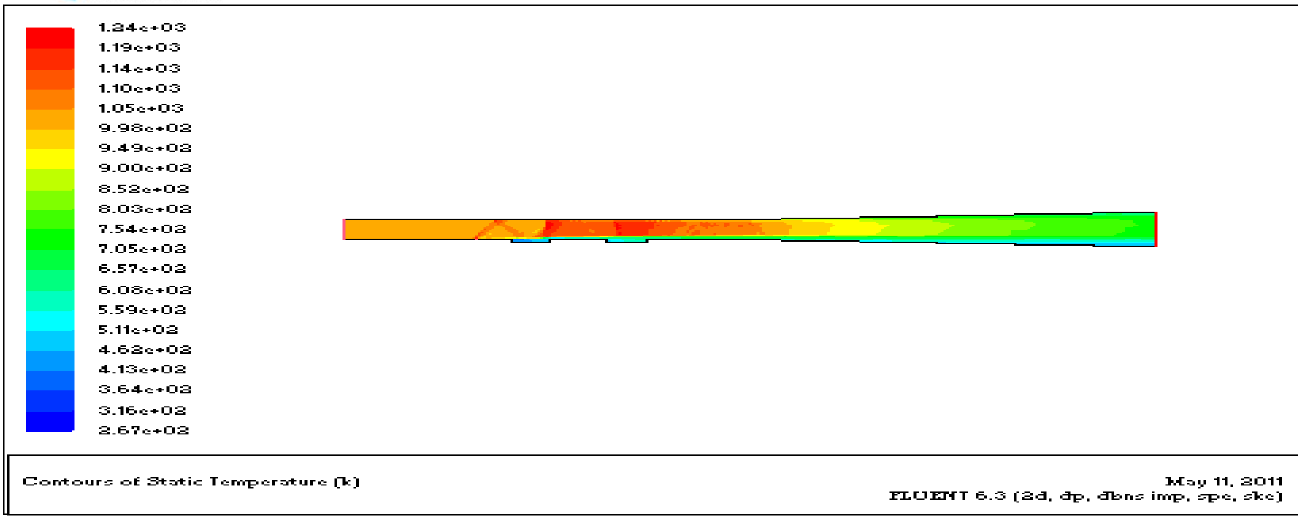

Fig.6. Contours of static temperature (K)

6. Conclusion

From the above contours we can evolutes that the variation of the pressure and temperature along with kinetic energy of scram jet engine with single and double cavities. In the case of single cavity fuel inlet there is a lower mixing in between fuel and air that’s why the static temperature development is less than the case of two cavity fuel inlets. And also the static pressure value is more in the case of second, but the velocity magnitude is higher in the first case due to the lower static pressure. The values obtained using fluent shows that the variation of pressure and temperature with double cavity combustion chamber is approximately around 3.20 bar and 1250 K. The working pressure and temperature of the scram jet is also close to the values obtained using single combustion chamber but the mixing phenomena of air and fuel at Mach number 1.4 is better using double cavity compared to a single cavity.

References

[1] Waltrup, Paul J., Michael E. White, Frederick Zarlingo, and Edward S. Gravlin."History of Ramjet and Scramjet Propulsion Development for U.S. Navy Missiles." Johns Hopkins APL Technical Digest , Volume 18, 1997: 234-243.

[2] Thompson, Elvia, Keith Henry, and Leslie Williams. "Faster Than a Speeding Bullet: Guinness Recognizes NASA Scramjet." NASA. 20 June 2005. RELEASE : 05-156. <http://www.nasa.gov/home/hqnews/2005/jun/HQ_05_156_X43A_Guinness.html>.

[3] "HyShot Program Secures Place in Flight History." The University of Queensland in Australia. 16 Aug. 2002. http://www.uq.edu.au/news/index.html?article=3469

[4] Heiser, William H., David T. Pratt, Daniel H. Daley, and Unmeel B. Mehta. Hypersonic Airbreathing Propulsion. Washington, D.C.: American Institute of Aeronautics and Astronautics, 1994.

[5] Curran, Edward T., "Scramjet Engines: The First Forty Years." Journal of Propulsion and Power, Volume 17, No. 6, November-December 2001.

[6] Fry, Ronald S. "A Century of Ramjet Propulsion Technology Evolution." Journal of Propulsion and Power, Volume 20, No. 1, January-February 2004: 27-58.

[7] Abbitt, J. D., Segal, C., McDaniel, J.C., Krauss, R.H.,Whitehurst, R.B., “Experimental Supersonic Hydrogen
Combustion Employing Staged Injection Behind a Rearward Facing Step”, Journal of Propulsion and Power, 9, 472-479, 1993.

[8] Ben-Yakar, A., Hanson, R. K., “Cavity Flame Holders for Ignition and Flame Stabilization in Scramjets: An Overview”, *J.* Propulsion and Power, 17, 869-877, 2001.

[9] Davidson, D. F., Petersen, E. L., Rohrig, M., Hanson, R.K., Bowman, C.T., “Measurement of the Rate Coefficient of for M=Ar and N2 at High Pressures”, *26*th Symposium (Int'l.) on Combustion, 481-488, 1996.

[10] Dixon-Lewis, G., “Complex Chemical Reactions Systems. Mathematical Modelling and Simulation”, J. Warnatz, W. Jager, Eds., Springer-Verlag, Berlin, 1987.

[11] Frenklach, M., Wang, H., Rabinowitz, M. J.,”Optimization and Analysis of Large Chemical Kinetic Mechanisms Using the Solution Mapping Method - Combustion of Methane”, Prog. Energy Combust. Sci.,18, 47-73, 1992.

[12] Fuller, R. P., Wu, P., Nejad, A. S., Schetz, J. A.,"Comparison of Physical and Aerodynamic Ramps as Fuel Injectors in Supersonic Flow",

Journal of Propulsion and Power, 14, 135-145, 1998.

[13] Hartfield, R. J., Hollo, S. D., McDaniel, J.C.," Experimental Investigation of a Supersonic Swept Ramp Injector Using Laser Induced Fluorescenc", Journal of Propulsion and Power, 10, 129-135, 1994.

[14] Hindmarsh, A. C., Broen, P. N., Grant, K. E.,Lee, S. L., Serban, R., Shumaker, D. E., Woodward, C. S.,"SUNDIALS: Suite of Nonlinear and Differential/Algebraic Equation Solvers", *ACM* Transactions on Mathematical Software, 31, 363-396, 2005.

[15] Erdos, J.I., Scramjet testing in Shock-Heated Tunels, ISSW21, Paper 9030, 1997

[16] Heiser, W. H., Pratt, D. T., Hypersonic Airbreathing Propulsion, AIAA, Washignton DC, 1994, pp 22-26

[17] Kerrebrock, J. L. Aircraft Engines and Gas Turbines, second edition, MIT Press, London, 1992

[18] O'Byrne, S, (1997) Examination of Transient Mixing and Combustor Geometry in a Scramjet, Masters Thesis,

[19] Curran, E.T., Heiser, W.H., Pratt, D.T., Fluid Phenomena in Scramjet Combustion Systems, Annual Review of Fluid Mechanics, Vol. 28, 1996

[20] Fox, J.S., O'Byrne, S.O., Gaston, M.J., Houwing, A.F.P., PLIF and Shadowgraph Imaging of Supersonic Mixing and Combustion, 1997

[21] Doolan, M.C., (1997) Optimising the Combustor Geometry in a Supersonic Combustor Ramjet, Honours thesis, Engineering Department, Australian National University

[22] Rabbath, P.A., (1994) Investigation of supersonic combustion in a Scramjet by using flow visualisation and pressure measurements, Honours thesis, Engineering Department, Australian National University

[23] Fox, J.S., (1998) Ph.D. thesis, to be submitted.

[24] Gaston, M.J., Mudford, N.R., Houwing, A.F.P., A Comparison of Two Hypermixing Fuel Injectors in a Supersonic Combustor, AIAA Journal of Propulsion and Power, 1997

[25] Anderson, John D., Modern Compressible Flow, 2nd edition, McGraw-Hill, New York, 1990

[26] seminarprojects.net/tscramjet?pid=37232&mode...

[27] ushotsearches.com/mixing-in-scramjet~0.html

[28] seminarprojects.com › ... › Mechanical Seminar Topics

[29] www.citeulike.org/journal/els-00102180/page/8

[30] www.highbeam.com/doc/1G1-262343231.html

[31] core.kmi.open.ac.uk/display/2681261

[32] Review of Air Vitiation Effects on Scramjet Ignition and Flame holding **...**

[33] www.lw20.com/Article.aspx?wd

[34] journals.ohiolink.edu/ejc/search.cgi?q=keywords...

[36] www.ijesd.org/papers/105-D537.pdf

[37] pdfsb.com/cfd+combustion+chamber+analysis

[38] www.combustioninstitute-indiansection.com/.../...

[39] www.ijetch.org/abstract/210-A553.htm

[40] ezinearticles.com › Travel and Leisure ›Aviation Airplanes

[41] uk.ask.com/beauty/Scramjet-Works

[42] www.ask.com/questions-about/Scramjet-Works

[43] www.ijetch.org/papers/208-A549.pdf

[44] webtothumb.com/Scramjets.html

[45] www.keybusinessplan.com/subsonic/subsonic.html

[46] www.dtic.mil/dtic/tr/fulltext/u2/a445171.pdf

[47] issuu.com/melwin.a3/docs/scramjet-combustor-development

[48] www.docstoc.com/.../International-Journal-of-Engineering-Research-...

[49] A. P. Singh , Oveepsa Chakraborty and K. M. Pandey, CFD Analysis of Cavity Based Combustion of Hydrogen at Mach number 1.2" The 11th Asian International Conference on Fluid Machinery and the 3rd Fluid Power Technology Exhibition. Paper ID: AICFM_ES_002, November 21-23, 2011, IIT Madras, Chennai, India.